USER'S MANUAL OF JANUS FOR VECTOR5

by

Agneya Sunil Loya

Serhan Guner

Frank J. Vecchio

August 2015

© Copyright by A. S. Loya, S. Guner, F. J. Vecchio (2015)

TABLE OF CONTENTS

1. Introduction **2**

2. Loading a model **2**

3. Layout **3**

4. The Taskbar **4**

4.1 Menus **4**

4.1.1 File menu 4

4.1.2 View menu 4

4.1.3 Structure menu 5

4.1.4 Results menu 5

4.1.5 Window menu 6

4.2 Toolbar buttons **6**

4.2.1 Open model file button 6

4.2.2 Hotspot view button 6

4.2.3 Material view button 6

4.2.4 Default view button 7

4.2.5 Sectional layer view button 7

4.2.6 Sectional crack view button 7

4.2.7 Global frame deformation view button 8

a. Deformations 9

b. Cracks 9

c. Reinforcement bars and stresses 10

4.2.8 View Options button 10

4.2.9 Plotting controls 11

4.2.10 Navigation dialog button 12

4.2.11 Legend dialog button 12

4.2.12 Video dialog button 12

4.2.13 Text position controls 13

4.2.14 Footer width controls 13

5. View options dialog box **14**

1. INTRODUCTION

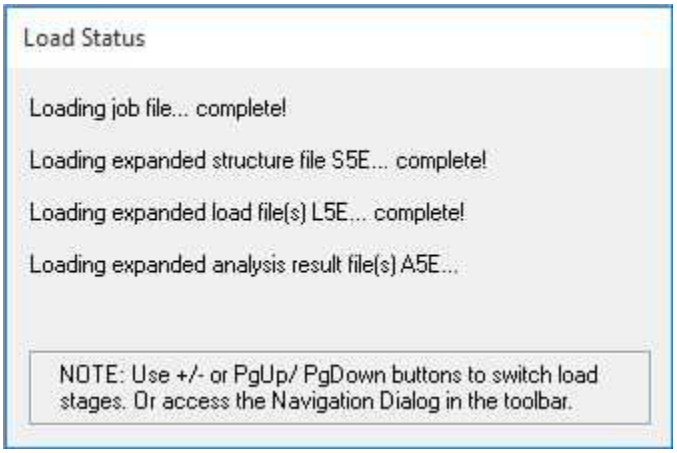
Janus is a post-processor for visualizing the analysis results of the VecTor suite of programs. This manual is specific to its use for VecTor5 models. Janus displays such information as the deformed shape, crack widths and positions, rebar positions and stresses and sectional views of members.

Janus is gradually developed and new futures are added. The following guide briefly summarizes the currently available features for VecTor5 models.

2. LOADING A MODEL

Loading a model is straightforward. Open 'Janus.exe' and browse for your VecTor.JOB file using the File -> Open button or the Open icon on the taskbar.

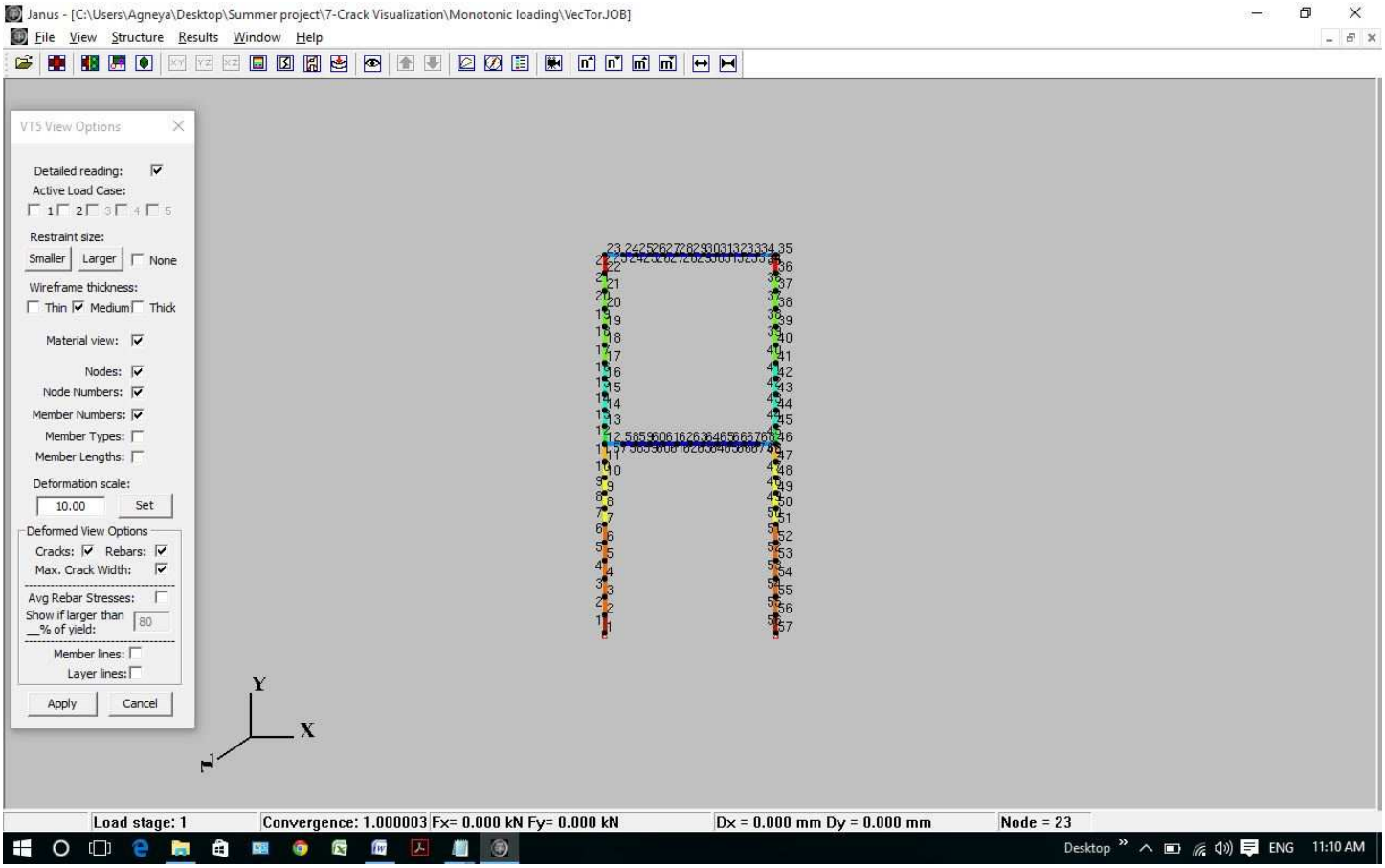
Janus displays the loading of the different model files with a pop-up box like this:


Load Status
Loading job file... complete!
Loading expanded structure file S5E... complete!
Loading expanded load file(s) L5E... complete!
Loading expanded analysis result file(s) A5E...
NOTE: Use +/- or PgUp/ PgDown buttons to switch load stages. Or access the Navigation Dialog in the toolbar.

New users should pay attention to the note in this box. You can **change load stages** by:

1. +/- buttons
2. PgUp/ PgDown buttons
3. Opening the navigation dialog box using the 'Compass' icon in the toolbar

Here is a snapshot of the screen after opening a model:



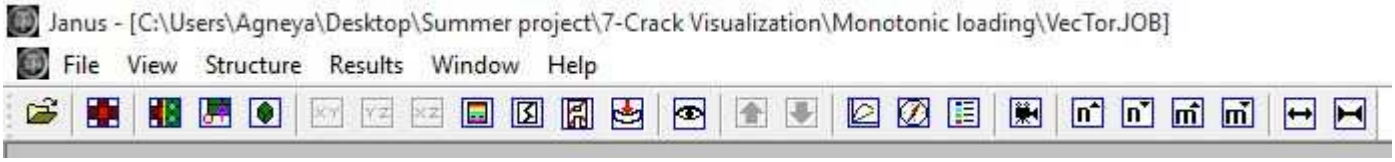
3. LAYOUT

All controls can be accessed through the taskbar on top. The status bar at the bottom contains indicators for (left to right):-

1. Load stage number
2. Convergence factor value
3. Cumulative applied force (force controlled) OR cumulative reaction developed (displacement controlled)
4. Displacement of node that undergoes max displacement
5. Node number of node that undergoes max displacement

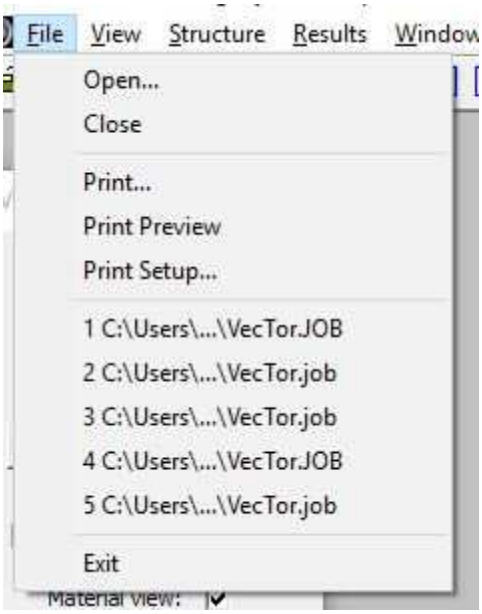
The VT5 View Options dialog opens by default on startup and contains the most important view controls for visualizing the model. It can be accessed using the ‘eye button’ in the taskbar.

4. THE TASKBAR


Janus - [C:\Users\Agneya\Desktop\Summer project\7-Crack Visualization\Monotonic loading\VecTor.JOB]
File View Structure Results Window Help

4.1 MENUS

4.1.1 File menu



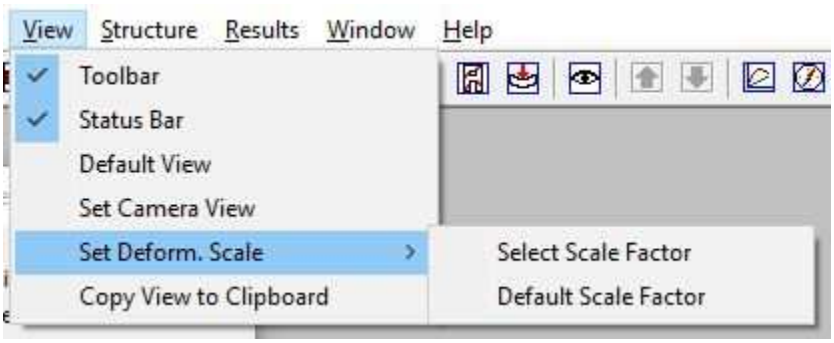
The file menu contains standard options to **Open** and **Close** files, **Print** the screen, **Open Recent** files and **Exit** the application.

4.1.2 View menu

The toolbar and status bar are active by default and can be hidden using the view menu.

The **Default View button** sets the model back to the view it initialized in.

Set Camera View provides controls to render the view from different angles and zoom levels. The only relevant control that the user may need is the **Set Zoom Scale factor** which allows finer or coarser control



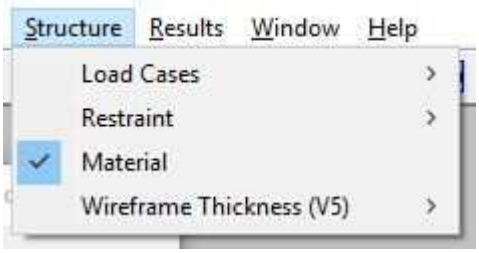
over the zooming in and zooming out of the model.

Set Deformation Scale has a submenu with options to enter a user defined value for the factor by which deformations are scaled and a button for the **Default Scale Factor** which is set to 10.0.

Copy View to Clipboard is for copying a snapshot of the screen to the clipboard. However, this button has bugs and the user is advised to use the default 'Snipping tool' provided by Windows.

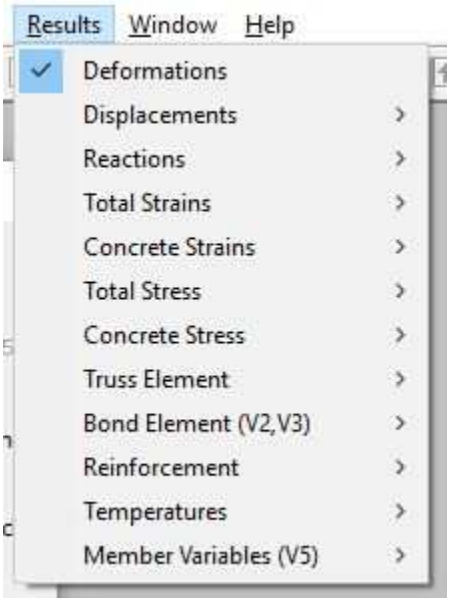
4.1.3 Structure menu

The structure menu contains controls to show or not show load cases, restraints, material colors and set the wireframe model thickness. However, the View Options dialog described later covers all these controls in a much more condensed and neat manner.

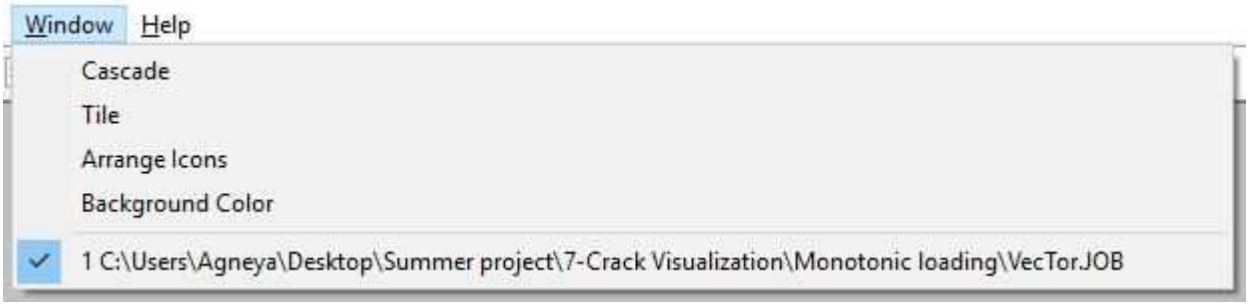


4.1.4 Results menu

Button	Function
Deformations	toggles the deformation of the model on and off
Displacements	x, y and z displacement gradients throughout the model
Reactions	x, y and z displacement gradients throughout the model
Total Strains	principal major and minor strain gradients
Concrete Strains	concrete strain gradients
Total Stress	**Not applicable for VecTor5**
Concrete Stress	Concrete stress gradients
Truss Element	**Not applicable for VecTor5**
Bond Element	**Not applicable for VecTor5**
Reinforcement	Long. and transverse reinforcement strain and stress gradients
Temperatures	**Not applicable for VecTor5**
Member Variables	Member forces and member deformations



4.1.5 Window menu



When multiple projects are open one can use the **Cascade** and **Tile** options to position the windows.

Arrange Icons is a redundant button

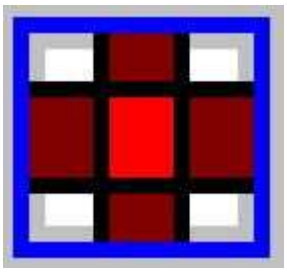
Background Color allows you to set the background color of the model view screen

4.2 TOOLBAR BUTTONS

4.2.1 Open model file button

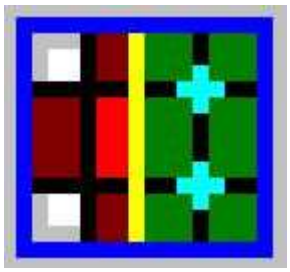
4.2.2 Hotspot view button

The hotspot view gives controls to view various structural parameters like deformations, stresses, strains and cracks as gradients on the model within a user defined range of values.



4.2.3 Material view button

Shows or hides colors to indicate different material types.

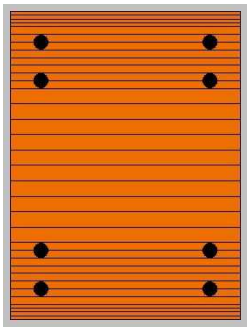


4.2.4 Default view button

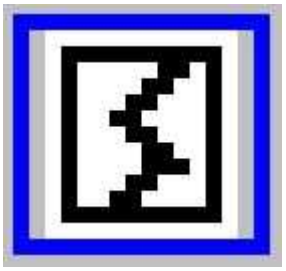
When no member cross-sectional view (described later) has been used this button returns the view to the same zoom level as initialization. When a cross-sectional view is active the button can be used to switch between the frame view and cross-sectional view.

4.2.5 Sectional layer view button

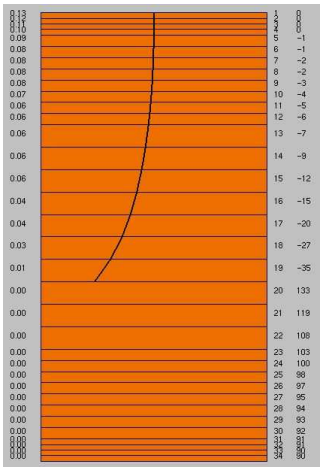
The sectional view button allows the user to select a member to see its cross sectional layer profile along with the reinforcement bar layout. An example of the sectional layer view is given below:



4.2.6 Sectional crack view button



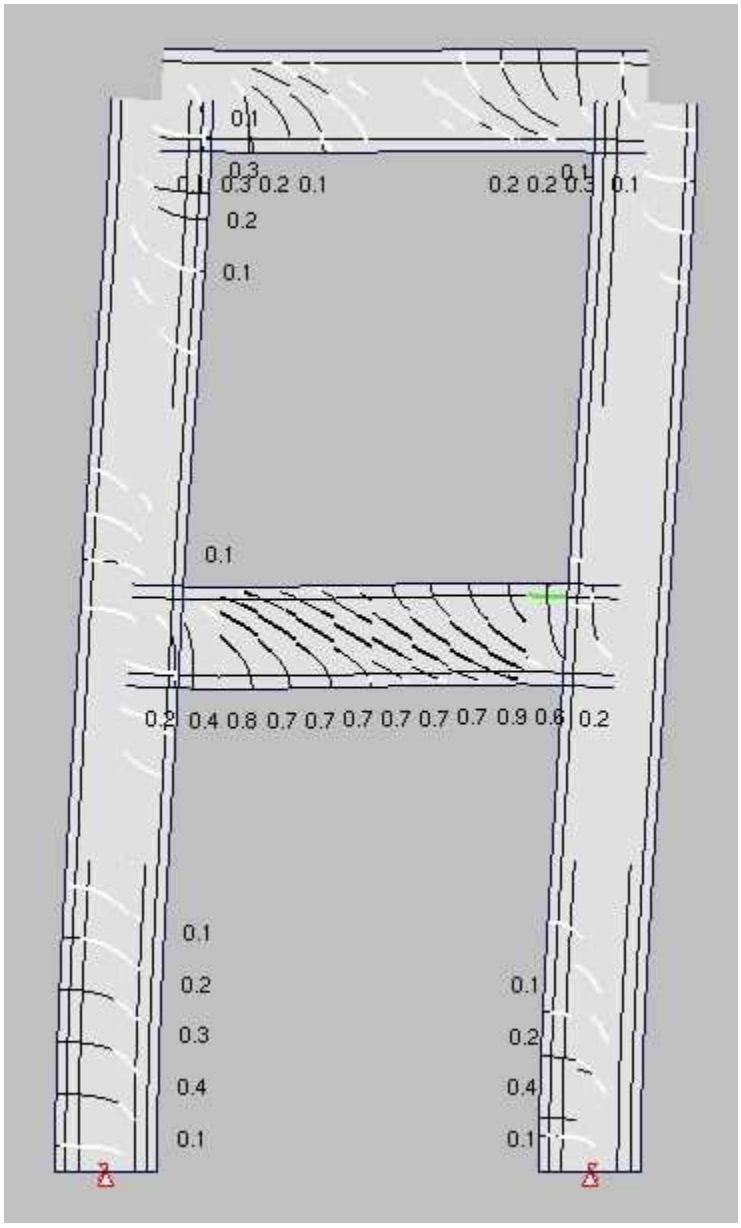
The sectional crack view button allows the user to select a member to see it cross sectional layer profile along with the computed cracking of the member. A sample of this view is given below. The list of numbers to the left of the layers is the value of crack width in each layer in millimeters.
The crack changes in response to changing load stages.



4.2.7 Global frame deformation view button

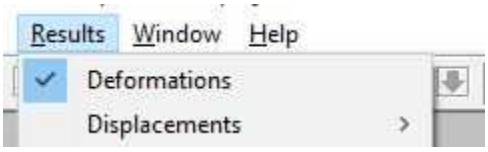
The frame deformation view shows the 2-dimensional profile of the frame taking into account deformations, cracking and reinforcement bar stresses.

Following is a sample view of a model in this mode:

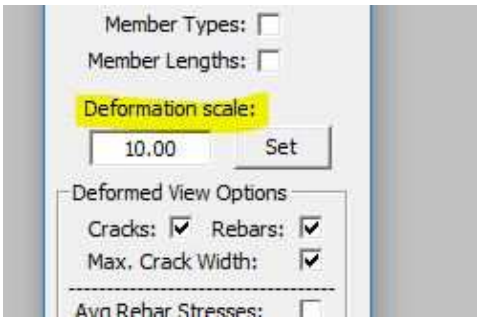


4.2.7 a) Deformations

The frame deforms by default with increasing load. The deformations can be switched off by clicking the Deformations button under the Results tab.



The calculated deformations are scaled to aid visualization. The default scaling factor for deformations is set to 10.0 and can be changed in the View Options dialog box.

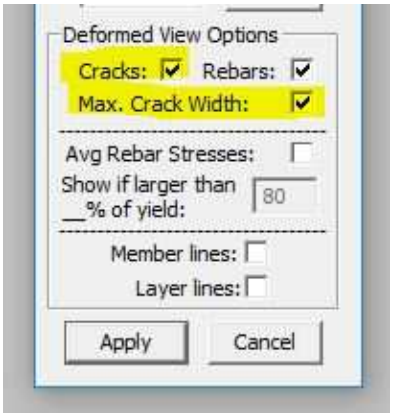


4.2.7 b) Cracks

Cracks are shown in different colors and thicknesses depending on their computed value.

Crack color and thickness	True physical crack width range
White + 2 pixels thick	< 0.10
Black + 1 pixel thick	0.10 – 0.50
Black + 2 pixels thick	0.50 – 1.00
Black + 3 pixels thick	1.00 – 1.50
Red + 2 pixels thick	1.50 – 3.00
Red + 3 pixels thick	> 3.00

The max crack widths per member are also printed along the figure. The cracks in the diagram and max crack width text can be shown or hidden using the View Options box.

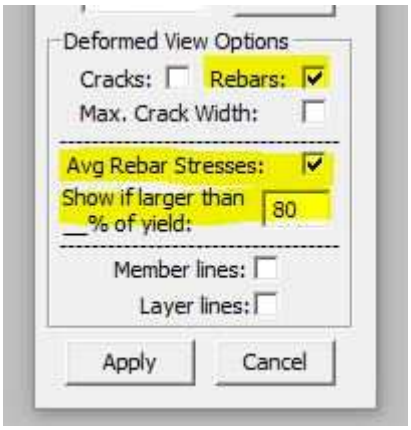


4.2.7 c) Reinforcement bars and stresses

Reinforcement bars are shown by default. The colour of reinforcement bars varies with their state. The key below summarises this feature:

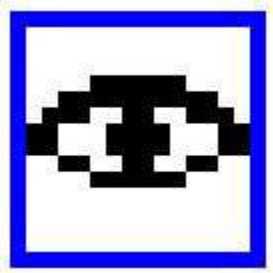
Rebar color	Rebar state
Black	Normal
Green	Yielding
Yellow	Strain hardening
Continuous Red	20% before rupture
Dashed Red	Post rupture

Reinforcement bars and their stress values can be shown or hidden using the View Options box. Rebar stress values are shown only when they are above a certain percentage of yield to avoid a cluttered view. This percentage is set as 80% by default and can be altered in the View Options box.

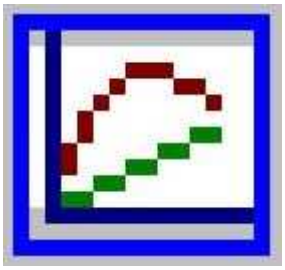


4.2.8 View Options button

This button opens the View Options dialog box. The dialog box is open by default. Details on the options contained in the dialog box are in section .

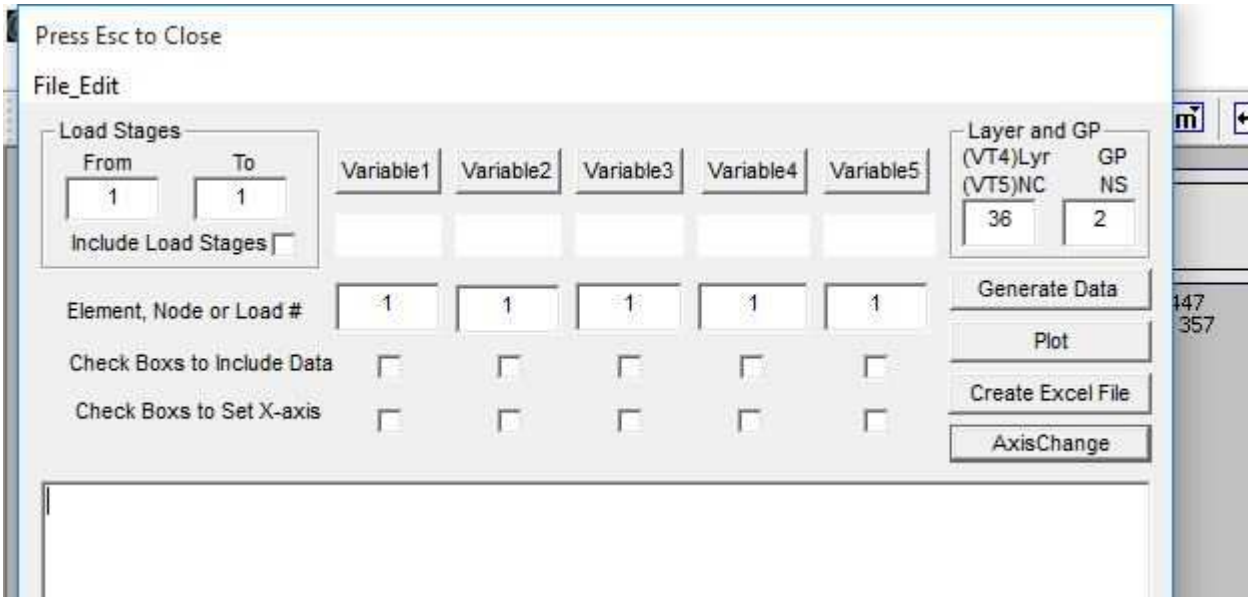


4.2.9 Plotting controls



The plotting controls button opens dialog boxes that allow the user to choose variables for plotting.

The first dialog box is:



To plot a graph:

1. Enter your load stage range
2. Select variables for plotting (usually 2 but can use up to 5)
3. Select the element (member) or node number for which you need the variable
4. Always check '*Check Boxs to Include Data*' corresponding to your selected variable
5. Choose the dependant variable to be on the axis by checking the box '*Check Boxs to Set X-axis*'
6. In the top left select NC (concrete layer number) and/or NS (steel layer number) for layer specific variables
7. Click plot
8. The plots created are not always very clear as this is feature is not completely ready. Clicking '*Generate Data*' generates the variable data for the load stage range in the white space below. This can be used to create your own graph in excel or a similar software

4.2.10 Navigation dialog button

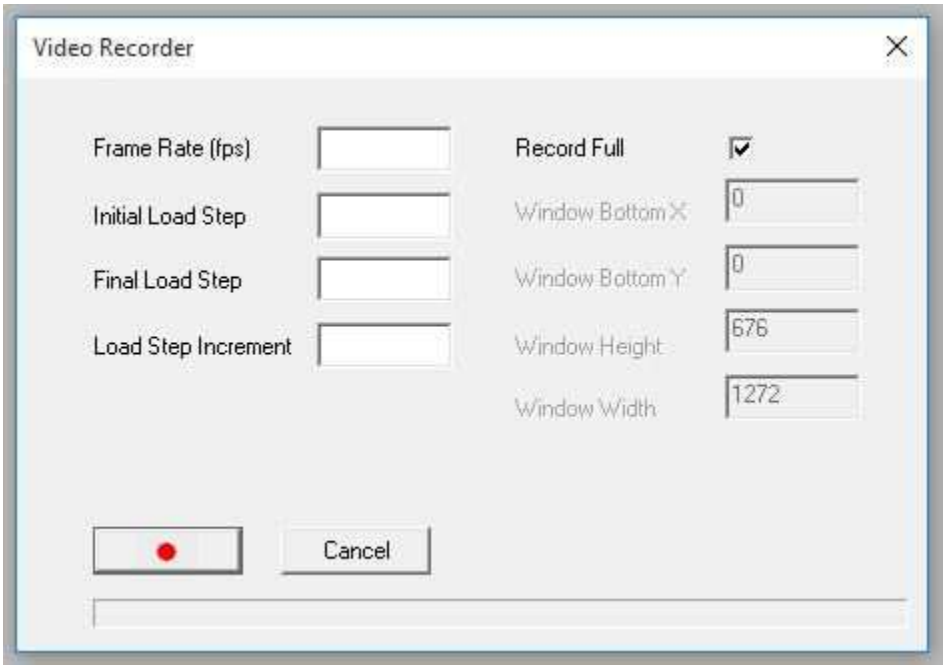
This opens the navigation dialog box. You may need it only if the +/- and PgUp/ PgDown buttons to switch load stages do not work for you.

4.2.11 Legend dialog button

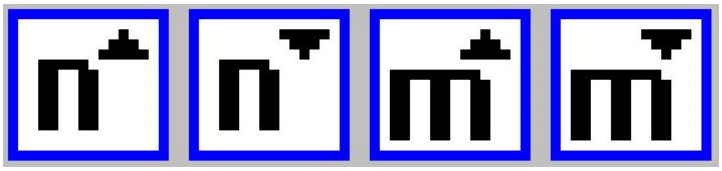
For material and gradient views the colours are presented in a legend dialog box. This button hides and shows the legend dialog box.

4.2.12 Video dialog button

This button shows the video dialog box (below). It can be used to record a video of the model deflection through increasing load stages. For a video at normal speed the advised frame rate is 24 fps. By default the video captures the whole window. To select a portion of the full window uncheck the '*Record Full*' box and specify the pixels you would like to capture. A guide box helps you choose the pixels as you change the values.

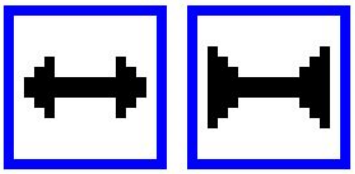


4.2.13 Text position controls



These **'n up'** and **'n down'** controls move the node numbers away or close to their respective nodes. The **'m up'** and **'m down'** buttons move the member numbers, member lengths, member types, crack widths and rebar stresses away or close to the model.

4.2.14 Footer width controls



These buttons control the width of the various status indicators in the footer. The footer is sized differently on screens with different resolutions and in some cases the data overlaps or spreads outside the frame. Resize the footer as per your convenience ensuring that you see a view as below:

5. VIEW OPTIONS DIALOG BOX

Whenever a setting in the view options dialog box is changes the apply button must be clicked for the changes to take effect.

S. No.	Button/ feature	Function
1	Detailed reading	Enables reading of detailed member outputs. Janus is fast enough to handle large models and this option can be left checked.
2	Active load case	Automatically enables boxes for up to 5 unique load cases. The user can view the action of only one load case at a time. However, note that the deformations and results are under the combined effect of all loads. Only point loads can be visualized.
3	Restraint size	Restraints can be made smaller, larger or hidden
4	Wireframe thickness	Changes the thickness of the model in the line diagram view
5	Material view	Shows or hides colors denoting different materials
6	Nodes	Shows or hides nodes
7	Node numbers	Shows or hides node numbers
8	Member numbers	Shows or hides member numbers
9	Member types	Shows or hides member types abbreviated MT__
10	Member lengths	Shows or hides member lengths in mm
11	Deformation scale	Displays current scale value and allow the user to set a new value. Enter a scale value between 0.1 and 100 and click 'Set'
12	Cracks	Shows or hides cracks
13	Rebars	Shows or hides reinforcement bars
14	Max. Crack Width	Shows the max crack width per member in mm
15	Avg Rebar Stresses	Shows the average rebar stress per member
16	Show if larger than _% of yield	Allows the user to specify cutoff value above which rebar stresses are shown
17	Member lines	Shows or hides member lines in frame view
18	Layer lines	Shows or hides layer lines in frame view